



Open Source CNC Toolpathing

Workflow

Written By: Chris Reilly

TOOLS:

- [Computer running Linux \(1\)](#)

SUMMARY

I'll demonstrate how to turn a raster image into a G-Code engraving pattern with GIMP, Inkscape, Pycam and EMC2 in Ubuntu Linux. The steps are covered in detail in this video:

Open Source CNC Toolpathing Workflow.

Step 1 — Open Source CNC Toolpathing Workflow

- Install the necessary software:
 - [Ubuntu Linux/EMC2](#)
 - [GIMP](#)
 - [Pycam](#)
 - [Inkscape](#)

Step 2

- In GIMP, open an image.
- Edit as you like, then apply a threshold (Colors->Threshold) to make a black-and-white only (not greyscale) image.
- Save the image.

Step 3

- In Inkscape, create a new file that reflects the size of your CNC work area.
- Import the image created in the previous step.
- Use the Trace Bitmap function (Path->Trace Bitmap) to create a vector outline of the raster image.
- Delete the imported raster image, leaving only the vector tracing.
- Save the file as an .SVG

Step 4

- Open the SVG in Pycam.
- Edit Pycam preferences:
 - Under General, set the units to inches.
 - Under Gcode, set the safety height to 1 inch.
- Adjust the tool parameters for the default cylindrical tool.
 - Diameter = 0.125"; Feed Rate = 75 inches/minute (this may vary depending on your CNC)
- Under Tasks, activate only the Gravure/Tracing task. Make these adjustments:
 - Tool = Cylindrical; Process = Gravure/Tracing

Step 5

- Generate the G-Code by clicking Generate Toolpath.
- Under Toolpaths, export as an .ngc file.
- Your G-Code is ready to run!

This document was last generated on 2012-11-02 04:25:43 PM.